

CATIA Syllabus

Module 1: Introduction to CATIA

• Overview of CAD and CATIA

- Introduction to CAD (Computer-Aided Design) and its significance in various industries.
- Introduction to CATIA: Overview of features, applications, and versions.
- CATIA interface: Menu bar, toolbars, specification tree, and 3D workspace.
- Understanding CATIA's role in Product Lifecycle Management (PLM).
- CATIA File Structure
 - CATIA file types: Part (.CATPart), Assembly (.CATAssembly), and Drawing (.CATDrawing).
 - Understanding CATIA
 Workbenches: Part Design,
 Assembly Design, Sheet Metal,
 Drafting, Generative Shape
 Design, etc.
- Navigating CATIA Environment
 - Basic navigation techniques in 3D space.
 - Understanding View Orientation, Zoom, and Pan in CATIA.

Module 2: Part Design

- Basic Part Modeling
 - Creating basic 3D parts from sketches: Points, lines, circles, rectangles, etc.
 - Using geometric constraints and dimensions for precise sketching.

- Creating 3D features: Extrude, Revolve, Pad, Pocket, Shaft, and Groove.
- Advanced Part Modeling
 - Complex 3D modeling features: Loft, Sweep, Rib, and Hole Wizard.
 - Working with fillets, chamfers, and drafts to refine part geometry.
 - Editing and modifying features, creating patterns, and using mirror tools.
- Working with Sketches
 - Creating and editing sketches: Adding dimensions, relations, and constraints.
 - Creating and using reference planes for 2D sketching.
 - Best practices for parametric design.
- Datum Features
 - Creating and working with datum planes, axes, and points.
 - Using reference geometry to build more complex parts.

Module 3: Assembly Design

- Creating Assemblies
 - Introduction to Assembly Design workbench.
 - Inserting parts and defining assembly constraints: Mate, Coincidence, Contact, Angle, etc.
 - Assembling components and managing parts in the assembly. Assembly Constraints and
 - Relationships
 - Working with assembly constraints: Fixed, Rigid,
 - Flexible, and Hybrid components.
 - Defining motion in an assembly.



• Managing large assemblies with a focus on performance optimization.

Assembly Features

- Creating assembly-based features like holes, cuts, and patterns.
- Understanding and working with subassemblies and component positions.
- Assembly Simulation
 - Simulating motion and movement between components.
 - Running interference detection to check for assembly conflicts.
 - Using kinematic analysis for mechanism design.

Module 4: Sheet Metal Design

• Introduction to Sheet Metal Workbench

- Overview of sheet metal design principles.
- Working with sheet metal features: Base, Flange, Edge Flange, and Bend.
- Creating parts with the "Unfold" tool to generate flat patterns.
- Advanced Sheet Metal Features
 - Using corner relief, hem features, and contour flanges.
 - Adding holes, cuts, and emboss features to sheet metal parts.
 - Flattening and analyzing sheet metal parts for manufacturing.

Creating Sheet Metal Drawings

- Generating technical drawings for sheet metal parts, including flat patterns.
- Using bend tables and creating custom annotations for sheet metal designs.

Module 5: Surface Design

- Introduction to Surface Modeling
 - Overview of surface modeling and its importance in complex product design.
 - Working with tools like Extrude, Sweep, and Loft to create surfaces.
 - Creating boundary surfaces and advanced filleting.

Advanced Surface Modeling

- Using advanced surface tools: Generative Shape Design (GSD), Multi-Section Surface, and Fill Surface.
- Creating and manipulating NURBS (Non-Uniform Rational B-Splines) surfaces.
- Combining surfaces and solids to create complex designs.
- Surface Analysis
 - Checking surface continuity, tangency, and curvature using analysis tools.
 - Repairing and optimizing surfaces for manufacturability.

Module 6: Drafting and 2D Drawings

Creating 2D Drawings from 3D Models

- Introduction to the Drafting Workbench in CATIA.
- Creating views from 3D models: Orthographic views, section views, and isometric views.
- Generating auxiliary and detail views to illustrate specific parts of the model.
- Dimensioning and Annotations
 - Adding dimensions: Linear, angular, radial, and ordinate.



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- Applying geometric tolerances, surface finishes, and callouts.
- Creating Bill of Materials (BOM) for assemblies.
- Advanced Drawing Techniques
 - Using view manipulation tools: Break views, projection, and exploding views for assemblies.
 - Creating title blocks, borders, and custom drawing templates.
 - Managing layers and drawing sheets for better organization.

Module 7: Product Data Management (PDM)

- Introduction to PDM and PLM
 - Understanding the role of Product Data Management in the design and manufacturing process.
 - Managing data and configurations within a Product Lifecycle Management system.
- CATIA and ENOVIA Integration
 - Using CATIA with ENOVIA for data management, collaboration, and version control.
 - Managing parts, assemblies, and drawings in a collaborative environment.
- PDM in Design and Manufacturing
 - Revision control, change management, and tracking product life cycles.
 - Integrating CATIA with ERP systems for design-tomanufacturing flow.

Module 8: CATIA Simulation and Analysis

Introduction to CATIA Simulation

- Overview of CATIA Simulation Workbench.
- Static structural analysis: Applying loads, boundary conditions, and materials.
- Running simulations to evaluate stress, strain, and displacement.

Advanced Analysis Techniques

- Modal analysis: Evaluating natural frequencies and mode shapes.
- Thermal analysis: Evaluating temperature distribution and thermal effects.
- Fatigue and vibration analysis for product testing.
- Optimization and Performance
 - Using CATIA optimization tools to enhance the design for strength, weight, and performance.
 - Interpreting results and refining the design based on simulation feedback.

Module 9: Generative Design and Additive Manufacturing

- Introduction to Generative Design
 - Principles of generative design and its applications.
 - Setting up generative design parameters and running optimization.
 - Analyzing results and refining the design for lightweight structures.
 - Additive Manufacturing (3D Printing)
 - Preparing CATIA models for additive manufacturing.
 - Understanding the different types of 3D printing and their applications in manufacturing.
 - Converting CATIA models to STL files for 3D printing.



Module 10: Final Project

• Hands-On Final Project

- Apply the skills learned to design a complex product, including part modeling, assembly, surface design, and technical drawing creation.
- Incorporating advanced features like sheet metal, surface modeling, and simulation.
- Presenting the final design with technical drawings, simulations, and analysis.

Project Review and Presentation

- Project review by instructors and peers.
- Demonstration of the design process, product analysis, and final project deliverables.

Tools and Technologies Covered:

- **CATIA Part Design**: Core tools for 3D solid modeling and parametric design.
- **CATIA Assembly Design**: Tools for creating and managing assemblies.
- **CATIA Sheet Metal Design**: Sheet metal modeling tools for creating flat patterns and bends.
- CATIA Generative Shape Design (GSD): Surface modeling tools for creating complex shapes.
- **CATIA Drafting**: Drawing creation tools for technical and engineering drawings.
- **CATIA Simulation**: Tools for structural, thermal, and fatigue analysis.
- **PDM/PLM Systems (ENOVIA)**: Data management and collaboration tools.

Learning Outcomes:

By the end of this course, students will be able to:

- Master CATIA's tools for 3D part modeling, assembly design, and surface modeling.
- Use CATIA for sheet metal and advanced surfacing techniques to create complex geometries.
- Create detailed technical drawings and manage product data using PDM systems.
- Perform structural, thermal, and vibration simulations to validate designs.
- Work with generative design and prepare models for additive manufacturing.